

How to 3D Print Your Brain

Get an MRI of your brain

- Steps

1. To do this, find a psych department in a nearby college and volunteer for an fmri experiment. Ask them ahead of time for your brain image - there might be HIPPA or IRB rules that could prevent them giving you the images so find out. Tell them you want your structural scan, and that dicom format is preferable.

Reconstruct the cortical surface of your brain

Note: These instructions, though adapted for our lab environment, were really put together by [this guy](#).

- Scripts & Files You Will Need For This Section

- freesurfer: recon-all

- Steps

1. You need to [download and install freesurfer](#). You will interact with freesurfer through Terminal (on a mac or linux machine) or command prompt on a Windows. Mac was used here. Mac or Linux are recommended. Follow the freesurfer wiki for an installation guide.
2. Add the freesurfer module. You can add this to your bash_profile if you want to use it always and forever in all terminals.

- module add freesurfer/5.3.0

1. Locate the folder where your brain images are. You need to know that file path for the next part.
2. In terminal, set up the reconstruction of your brain's cortical surface! This might take a while... Use the code below. After -i you need to put the file path to the first dicom image of your brain. You will also specify a name for a new folder freesurfer will create with your reconstructed brain image. Mine is called MR101 below.

- recon-all -i ./02+t1mprage/methisisme+02+t1mprage+00001.dcm -s MR101 -sd ./ -all
 - where:
 - -i takes the first image of the structural .dcm
 - -s is some made up subj ID
 - -sd is the folder you want the reconstructed files to go in
 - -all just comes at the end, I'm not sure why. Someone google why.
 - for more information: <http://ftp.nmr.mgh.harvard.edu/fswiki/recon-all>

Convert the reconstructed image into .stl

Motivation: You want it in a file format 3D printer's understand.

- Scripts & Files You Will Need For This Section

- freesurfer: mris_convert

- Steps

1. cd to the directory where your reconstructed images are (this file path will be specific to you. You want to cd to the new folder freesurfer just created).
 - cd /projects/3DPrint/[NAME]/surf
1. Now call this command for each hemisphere:
 - mris_convert lh.pial lh.pial.stl
 - mris_convert rh.pial rh.pial.stl

Downsample your .stl file in Meshlab

Motivation: In order to help prevent your printer from crashing, reduce the size of the image to have less than 20,000 faces.

- Scripts & Files You Will Need For This Section
 - [Meshlab](#)
- Steps
 1. Open meshlab and load your stl file
 1. Open --> Import Mesh --> select for R & L hemisphere stl files and import
 2. If it asks about merges vertices say yes
 2. Filters --> Remeshing, Simplification, and Reconstruction --> Quadric Edge Collapse Decimation
 1. Target Number of Faces: 19999 (or smaller)
 2. hit Apply, close window
 3. Save new file:
 1. Export Mesh As... save

Slice Your Brain, Make Printer Ready

Motivation: When you 3D print something, you want to make sure it has a stable structure (like won't topple over mid print) and that internally it has a stable structure to keep it from caving in). Programs do this for you. You also need to translate the files into G-Code which is what printers read. Please follow the instructions for your 3D Printer! You will use a slicing program to cut the model into thin horizontal segments. Like [Slic3r](#) or Cura or whatever is recommended by your 3D printer. The slicing program generates the tool path based on the characteristics of the model and your extrusion settings. This program will generate the G-Code that drives the printer. You then pass that file to a program that controls and operates the printer, like [Printrun/Pronterface](#).

Additionally, you can use an option in Slic3r or Cura called "Brim" to extrude a thin, one layer surface to help the edges of the print adhere to the print bed.

- Scripts & Files You Will Need For This Section
 - [Slic3r](#) or Cura
 - [Printrun/Pronterface](#) (not needed with Cura)
- Steps for Cura
 1. Launch Cura.
 2. Load your model into lulzbot (button on upper left of Cura)
 3. You can rescale
 4. select material you want to print with
 5. hit control

6. set head to 210 degrees (for ABS, PLA, etc) research temp for filaments
7. [How to change filament](#)
8. Hit print

Viola!







Guide by [Marianne Reddan](#)

www.appliedmarianne.com